INJECTION MOLDING METHODS DESIGN, OPTIMIZATION, SIMULATION OF PLASTIC BOTTLE CAP PART BY MOLD FLOW ANALYSIS

Vijaykumar Vilas Andhalkar[1], Dr. S. R. Dulange[2]
[1] Department of Mechanical Engineering, P.G Student,
A.G. Patil Institute Of Technology, Solapur
[2] Department of Mechanical Engineering, Associate Professor,
A.G. Patil Institute Of Technology, Solapur

Abstract
In this paper a component referred to as bottle cap where chosen for a comprehensive design review and mould flow analysis. The design of this product and the mould were made by the designing analysis software Autodesk Inventor software, which is then simulated by the use of Autodesk Mold Flow software. The mold flow analysis is used to predict the deformation of the part, and then adjust the design accordingly and this is done using the Mold-Flow system.
Key Words: Injection moulding, Mould design, Mold flow simulation, Optimization Plastic Injection mould, Mould Flow Plastic Advisor (MPA)

1. INTRODUCTION

Injection moulding is one of the most important processes in the plastic manufacturing industry. More than one-third of all plastic materials are injection molded, and the mold is one of the main components in the injection molding process.

The Autodesk Simulation Mold flow results help to identify the main problem areas before the part is manufactured that are particularly difficult to predict with traditional methods.

Analysis is essential for designing and mould making through simulation step-up and result interpretation to show how changes to wall thickness, gate location, material and geometry affects manufacturability and also experiments with "what-if" scenarios before finalizing a design. Injection Moulding simulation software into the mould design process in order to analyze the product, foresee the possible defects, and optimize the design to achieve the maximum outcome of the products with minimum cycle time in each production cycle.

2. PROBLEM DEFINITION:

This work related to injection molding problems of bottle cap plastic part of Kankaria plastic industries at Barshi Dist. solapur.India . The part Bottle cap have Flash problem compony wants to resolve it or modify in mould die . So, here the material flow into two -plate molds will be analyzed by using mold flow adviser.

Fig2A.Defects in pressure reducer Part

3. OBJECTIVES OF THE WORK

1. To analyze the behavior of Thermoplastic material during the production cycle from the filling phase until the ejection phase
2. To foresee the possible problem for a product design; and therefore able to op-timize the design in the mould design process
3. To achieve the minimum production cycle time
4. To construct a rapid prototyping of the mould cavity design into a standard aluminum or steel mould plate

5.To prepare a product design for " reducer plastic part by using design analysis software
6. Using Mold Flow to simulate the polymer flow and finding out maximum clamp force and Fill time

4. Model details

A 3d model of part pressure reducer is created in Autodesk Inventor software

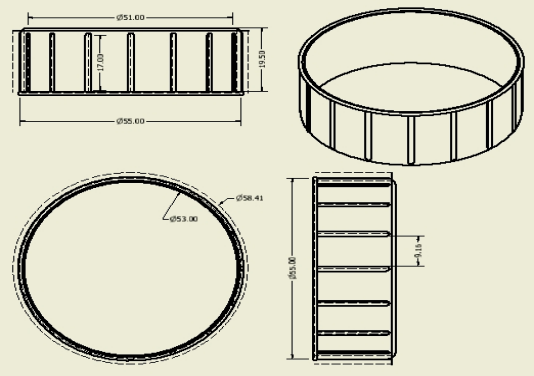


Fig.4A)CAD Models For Pressure Reducer

Fig 4B) Mould For Existing bottle cap plastic part

5. Process settings

Melt temperature: 220 (C)
Mold temperature: 50 (C)
Injection locations: 1
Max. machine injection pressure: 22.48(MPa)

6. Material Specification

Family Name: POLYETHYLENES (PE)
Trade Name : Generic HDPE
Family Abbreviation : HDPE
Material Structure : Crystalline
Mechanical Properties
Basic Modulus 911Mpa
Poisson's Ratio : 0.426
Shear Modulus : 319.4 Mpa
Mold Temperature Range 2 0-70 °C
Melt Temperature 220°C

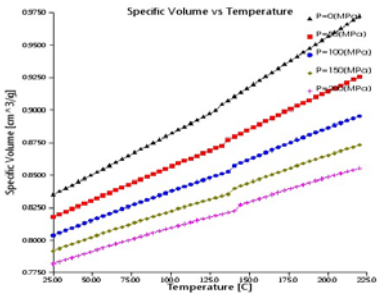


Fig.6a) Mechanical properties of HDPE

7. Simulation Result

7.A Gate analysis Result

Optimum gate locations may need to be examining by running the filling analysis on different best gate locations. Figure shows the result of gate location. Blue area represents the best gate locations for the part.

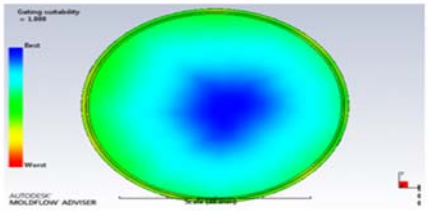


Fig.7. A) Best Gate Location

7.b)Fill Analysis Result

The Fill time result shows the position of the flow front at regular intervals as the cavity fills. At the start of injection, the result is dark blue, and the last places to fill are red. If the part is a short shot, the section which did not fill has no colour. Fill time is the time taken to fill up the part inside the cavity; it is also to show how the plastic material flows to fill the cavity

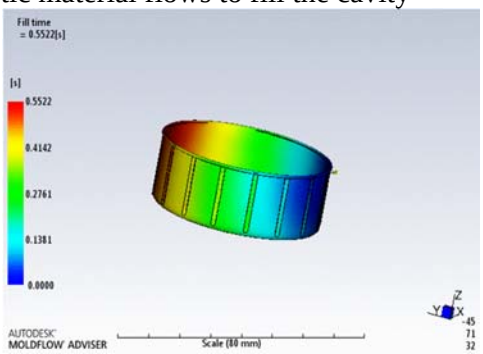


Fig.7b A) Fill time analysis result for old process setting parameters

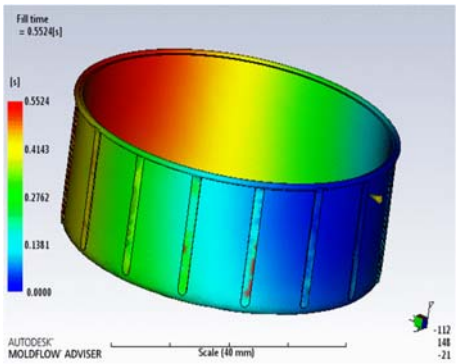
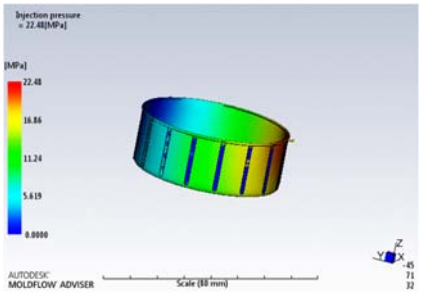


Fig.7b) B Fill time analysis result for NEW process setting parameters

7C)Injection pressure Analysis result

The analysis is considered with an injection machine of maximum pressure 220 MPa and operating pressure of 27MPa as a start basic setting of the analysis. The result obtained from the simulation of the analysis shows that though a maximum pressure in the red zone is allowed as high as 22 MPa that would be enough for complete filling.



7C)A Injection pressure analysis result for old process setting paramet

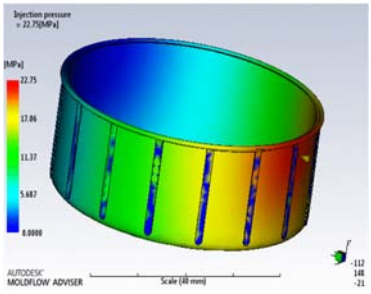


Fig. 7C)B Injection pressure analysis result for New process setting parameters

7D)Confidence of Fill Analysis result

The confidence of fill result from the plastic filling analysis displays the probability of plastic filling a region within the cavity under conventional injection molding conditions. If the cavity does not fill (short shot), the changes must be made to the processing conditions, injection locations, design of the parts, or choice of the plastic. Table shows the risk of the part filling base on colors.

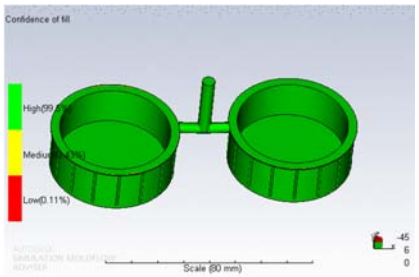


Fig. 7D) A }Confidence of filling analysis result for Old process setting parameters

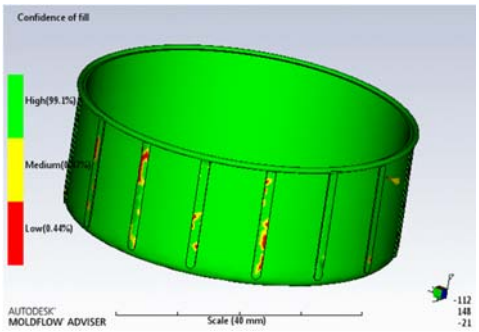


Fig. 7D) B}Confidence of filling analysis result for New process setting parameters

7E)Flow Front Temperature Analysis result

If the flow front temperature is too low in a thin area of the part, hesitation or short shot may be occurred. If it is too low in an area where weld lines are present, the weld lines may appear worse.

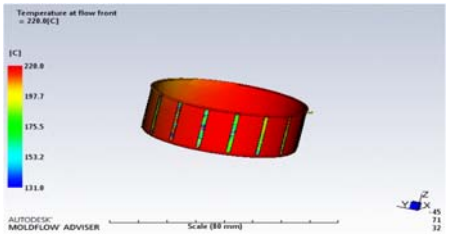


Fig. 7 E) A }Temp at flow front Old methodology

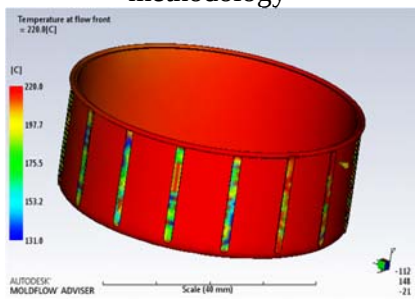


Fig. 7 E) B }Temp at flow front New methodology

8. Prototyping of the Mould

Finally, after the core and the cavities designed and the mould aggregate achieved the next step can be employed for the prototyping simulation using Mastercam in which the design will be imported from Autodesk in the extension file of IGES/STL . Once imported, the pocket tool path functions needs to be used for milling the product as seen on figure .CNC Program and tool path simultation is carried out by MasterCAM software. At the end the G code can be generated with this software which feeds directly in to the CNC machine for for facing mating surfaces of cavity die plate 222 and core plate 333 material steel.

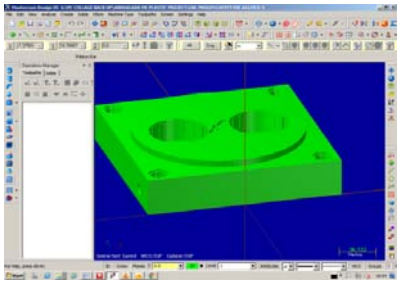


Fig8.a)Machine toolpath setting cavity die plate

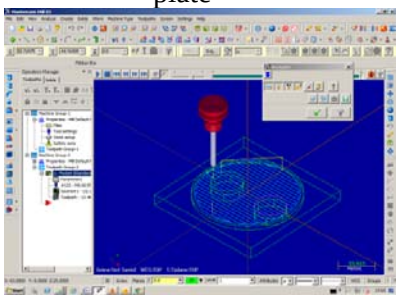


Fig8b). verification of the tool path core plate 333 in the MasterCAM

9. Mould Parts Manufacturing

EDM machine is used to drill hole cavity die plate 222 for shifting gate location to downward. EDM machining is a machining method to Machine hardened steel and other difficult materials

Fig9.Modified Gate Location by drilling Hole By EDM Machine

10. RESULTS AND DISCUSSION

Table -1: Mold Flow analysis Result Table for bottle cap part

Result	OLD PROCESS PARAMETERS	NEW PROCESS PARAMETERS
Material trade name:	Generic HDPE	Generic HDPE
Melt temperature	220	220
Mold temperature	50	50
Injection Locations per cavity	1	1
Fill Time	0.552 S	0.554 S
Actual injection pressure:	22.48 Mpa	22.75Mpa
Clamp force area	24.9209 (cm^2)	24.9309 (cm^2)
Max. clamp force during filling	2.518 (tonne)	2.509 (tonne)
Estimated cycle time:	: 7.60 (s)	7.70 (s)
Velocity/pressure switch-over at time:	0.54 (s)	0.54 (s)

The comparison of initial and modified designs on various parameters for bottle cap part

1) Gate location is an important and critical area in plastic injection molding and in die designing. An optimum gate location is always the desired by the mold maker while making a mold for any component. Gate position and the best one from the available is always found or taken into account based on the mold maker's experience

2)The results clearly describes the optimum gate position for different scenarios based on which a mold can be fabricated and a gate can be fixed.

3)If Gate location for bottle cap part changed to lower position of existing gate then flash problem of bottle cap avoided.

4) After comparison of results of existing and modified gate location for bottle cap part .The fill time ,quality of part filling for modified gate location is better than existing gate location and also no other defects like weld line ,shrinkage ,air trap visualize.

11. CONCLUSIONS

In this case study bottle cap have frequent problem in flash .The part is analyzed for existing gate location . The gate analysis helps to define the most suitable gate for the product; it does not necessarily have to be the best gate location but it has to be the most opti-mum use for the particular product .The gate location is changed to lower position of existing gate location. After comparison of results of existing and modified gate location .The fill time ,quality of part filling for modified gate location is better than existing gate location and also no other defects like weld line ,shrinkage ,air trap visualize .So in existing mould gate location is changed to lower position by drilling hole by Di-sinking EDM machine and surfaces of core plate and cavity die plate are faced by CNC Machines .With use of MASTERCAM software CNC Part program generated and simulated for surface facing operation of core plate and cavity die plate After this problem of flash is resolved for bottle cap .

In old gate location product a runner and gate system have to cut from part manually by labors but by modifying gate location, a runner and gate system automatically cut from part when part is ejected from mould cavity .This is additional advantages occurs and reduce production time.

REFERENCES

1. Auto desk Mold-Flow Insight material data warehouse.

2. wikimedia Foundation, Inc, 2010. InjectionMoulding. [online] Available at: <http://en. wikipedia. org/wiki/Injection_ moulding> [Accessed 28 August 2010]

3. Bryce, D. M., 'Plastic injection molding: Manufacturing process fundamental'. Society of Manufacturing Engineers, (1996)

4. Moldflow Plastic Insight, 2014. Moldflow Tutorial. [Software tutorial] Mold-flow Corporation

5. Tang, S.H., Kong, Y.M.,Sapuan, S.M., Samin, R., and Sulaiman, S., "Design and thermal analysis of plastic injection mold" , Journal of Materials Processing Technology, Vol. 171, pp. 259-267, 2006.

6.http://www.engineersedge.com/injection_molding,.htm, 2006

7.http://www.efunda.com/DesignStandards/plastic_design/plastic_intro.cfm, 2006

8. Mohd Amran, Siti Salmah, Zolkarnain Marjom, Umar Al-Amani, Raja Izamshah, Mohd Hadzley, Zulkeflee Abdullah, Mohd Sanusi "Warpage analysis verification between simulation and experimental of dumbbell plastic part in the injection moulding process" .International Symposium on Research in Innovation and Sustainability 2014 (ISoRIS '14) 15-16 October 2014, Malacca, Malaysia Special Issue Sci.Int.(Lahore),26(4),1575-1579,2014 ISSN 1013-5316; CODEN: SINTE 8

9. Thi Truc-Ngan Huynh "Using Injection Molding Process for Manufacturing a HDPE Mini Wheel" J. Eng. Technol. Educ. (2013) 10(4): 450-466 December 2013

10. Yousef Amer, Mehdi Moayyedian, Zeinab Hajiabolhasani, and Lida Moayyedian "Improving injection moulding processes using Experimental design" World Academy of Science, Engineering and Technology Vol:7 2013-03-28

11.Manmit Salunke, Rushikesh Kate, Vishwas Lomate, Gajanan Sopal "Injection molding methods design, optimization, Simulation of plastic toy building block by moldFlow analysis" Volume 6, Issue 6, June (2015), pp. 33-41 Article ID: 30120150606004International Journal of Mechanical Engineering and Technology © IAEME: http://www.iaeme.com/IJMET.asp ISSN 0976 – 6340 (Print) ISSN 0976 – 6359 (Online)

12. Hariyanto Gunawan and Willyanto Anggono **"** Improving quality of injection mold using moldflow software simulationcase study: new design plastic cup" ISBN 979-15577-0-5, proceeding of international seminar on product design and development 2006 yogyakarta-indonesia, december 13 – 14, 2006

13.H Adithya Bhat, Sandeep Subramaniam, Arvind K. Pillai, Edu Krishnan L, M. Elangovan" "Analysis and design of mold for plastic side release buckle using moldflow software" eISSN: 2319-1163 | pISSN: 2321-7308

14. All about plastic moulding. 2009. The History of Plastic Moulding. [ONLINE] Available at: http://www. Plasticmoulding .ca/ history.htm#Plastic. [Accessed 10 October 11].

15.Murti, B, 2010. Simulation and analysis of Injection Moulding product and rapid prototyping. Bachelor. Finland: Arcada University.